Abstract

Thermal stresses arise in a component that undergoes a change in temperature while the material is either externally or internally constrained, preventing it from freely expanding or contracting with the temperature change. The aim of thermal stresses analysis is to ensure the reliability of the design and reduce uncertainty in materials undergoing heat treatment. Thermal stress analysis involves examining the thermomechanical system, which is subjected to heat transfer and heat generation within the material (i.e., induction heating). Therefore, types of multiphysics analysis achieved through numerical methods are the appropriate approach. The study consists of two phases: controlling the heating parameters and subsequent cooling (quenching). Each phase has its own design strategy and controlling mechanism, and failure of one affects the desired mechanical properties.

A precise numerical model that describes the events on the surface of a workpiece during induction heating (IH) as well as the temperature changes within the workpiece is challenging to develop. To address this challenge, the model employs three different algorithms to analyse the temperature distribution from the surface to the core: explicit and implicit event controlling algorithms and a discrete frequency control approach aligned with the coil current. This is achieved through the automation of feedback control mechanisms that adjust the input parameters. Controlling the temperature distribution by adjusting the input parameters is regarded as a means of governing the entire system in induction heating.

The induction hardening model for gear wheels is calibrated to ensure high precision, based on a reliable mathematical model and the correct selection of input parameters. The calibration strategy integrates key parameters to achieve the most effective combination for the specified application, control heat flow, and ensure optimal energy efficiency. However, some of these parameters are often known only with some uncertainty (e.g., physical properties of the material and their temperature dependencies, parameters of cooling of heated teeth, etc.). Therefore, the model must be appropriately calibrated to achieve an acceptable agreement between the calculated results and the experimental data. The detailed calibration strategy is described and illustrated with a typical example.

Thermal residual stress resulting from volume changes due to temperature variations and phase transformations, and its effects on other mechanical properties, has been investigated. A general model of induction surface hardening was analysed on the basis of coupled electromagnetic, thermal, mechanical, and metallurgical phenomena. The distribution of mechanical strains and stresses is determined in the surface layers of steel materials subjected to induction hardening. This distribution is influenced not only by thermoelastic processes but also by plastic deformations of the exposed layers and the transformation of specific levels of steel. An illustrative example demonstrates the methodology to solve an axisymmetric configuration.

Extracting the transformation-induced plasticity (TRIP) strain from the total strain by experimental analysis of quenching poses significant challenges. However, the Finite Element Method (FEM) is crucial in addressing these challenges and enabling the prediction of multiple co-occurring physical phenomena. The combined effect of thermal and TRIP strains on mechanical properties is demonstrated through an illustrative example using a cylindrical workpiece with a length of 100 mm and a radius of 20 mm. The model also illustrates how phase transformation strains produce stresses and deformations by coupling temperature-dependent phase transformations with an elastoplastic analysis.

The main objective of this work is to minimise uncertainty in model development, optimise processes, and ensure energy efficiency. Uncertainty modelling is helpful in decision-making for new models, especially for processes involving multiphysics (i.e., induction hardening). In this work, a numerical model was developed using COMSOL Multiphysics and data analysis was performed using MATLAB software connected via LiveLink. Temperature control algorithms were developed to explicitly or implicitly define events in the heating process and control them via a feedback loop. Another method used involves defining an objective function subject to constraints and a certain group of parameters. This method is crucial to identify the most influential parameters in the process and to select optimal values.

Elastoplastic models of martensite formation have been presented, providing an overview of different modelling approaches related to martensite formation. In this approach, FEM is implemented to analyse residual thermal stresses, strain formation, and plastic strain evolves when deviatoric stresses and thermal stresses exceed the yield limit.